3D Printing low-run injection molds

Written by Ben Redwood

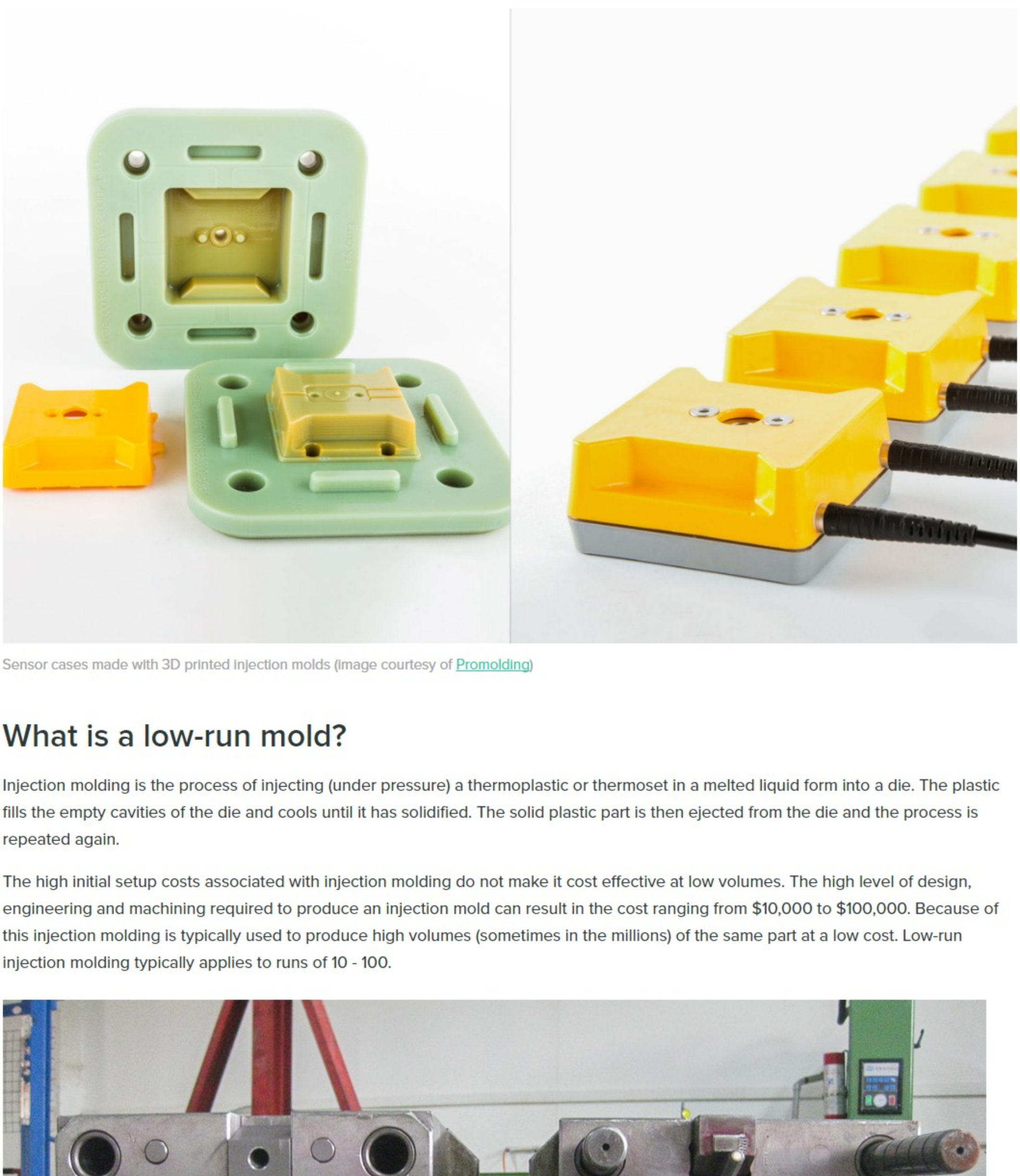
This article discusses the use of 3D printing to print molds for low run injection molding. Design considerations, materials, molds configurations and a comparative case study are all included

Table of contents

Introduction

Injection molding is the most common method for producing plastic parts. While traditionally 3D printing was only used for verifying prototypes of parts that were later going to be injection molded, developments in printer accuracy and materials now allow 3D printers to print injection molds directly.

This article will discuss the benefits of 3D printing low-run injection molds and give advice on the best mold configuration, mold materials and how to design a 3D printed injection mold.

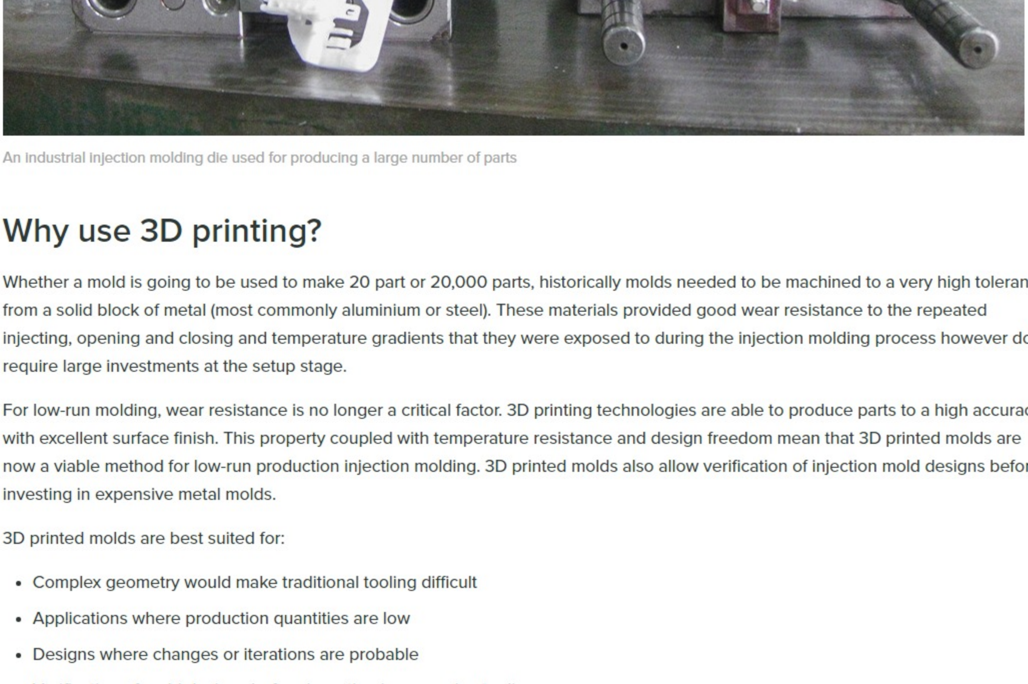


Sensor cases made with 3D printed injection molds (image courtesy of Promolding)

What is a low-run mold?

Injection molding is the process of injecting (under pressure) a thermoplastic or thermoset in a melted liquid form into a die. The plastic fills the empty cavities of the die and cools until it has solidified. The solid plastic part is then ejected from the die and the process is repeated again.

The high initial setup costs associated with injection molding do not make it cost effective at low volumes. The high level of design, engineering and machining required to produce an injection mold can result in the cost ranging from $10,000 to $100,000. Because of this injection molding is typically used to produce high volumes (sometimes in the millions) of the same part at a low cost. Low-run injection molding typically applies to runs of 10 - 100.



An industrial injection molding die used for producing a large number of parts

Why use 3D printing?

Whether a mold is going to be used to make 20 part or 20,000 parts, historically molds needed to be machined to a very high tolerance from a solid block of metal (most commonly aluminium or steel). These materials provided good wear resistance to the repeated injecting, opening and closing and temperature gradients that they were exposed to during the injection molding process however do require large investments at the setup stage.

For low-run molding, wear resistance is no longer a critical factor. 3D printing technologies are able to produce parts to a high accuracy with excellent surface finish. This property coupled with temperature resistance and design freedom mean that 3D printed molds are now a viable method for low-run production injection molding. 3D printed molds also allow verification of injection mold designs before investing in expensive metal molds.

3D printed molds are best suited for:

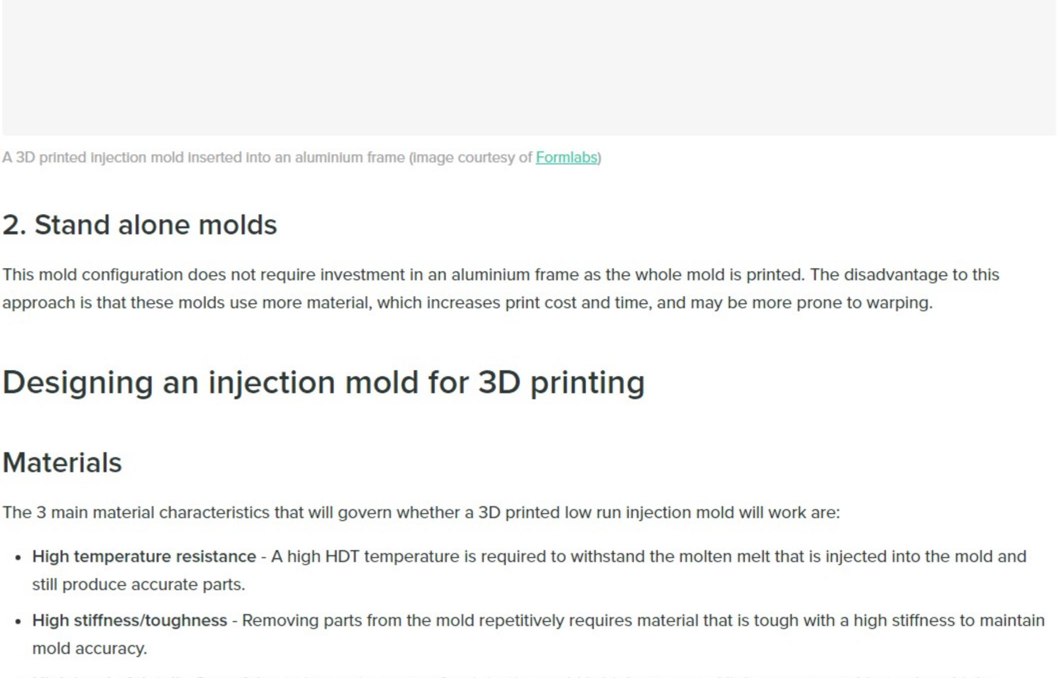
- Complex geometry would make traditional tooling difficult
- Applications where production quantities are low
- Designs where changes or iterations are probable
- Verification of mold designs before investing in expensive tooling

Mold configurations

3D printed injection molds are produced in 2 standard configurations.

1. Mold inserts for aluminium frames

This is the most common 3D printed mold configuration and generally produces more accurate parts. The mold is 3D printed and inserted into aluminium frames which provide support against the downward pressure and heat of the injection nozzle. Aluminium frames also help prevent the mold from warping after repeated usage.



A 3D printed injection mold inserted into an aluminium frame (image courtesy of Formlabs)

2. Stand alone molds

This mold configuration does not require investment in an aluminium frame as the whole mold is printed. The disadvantage to this approach is that these molds use more material, which increases print cost and time, and may be more prone to warping.

Designing an injection mold for 3D printing

Materials

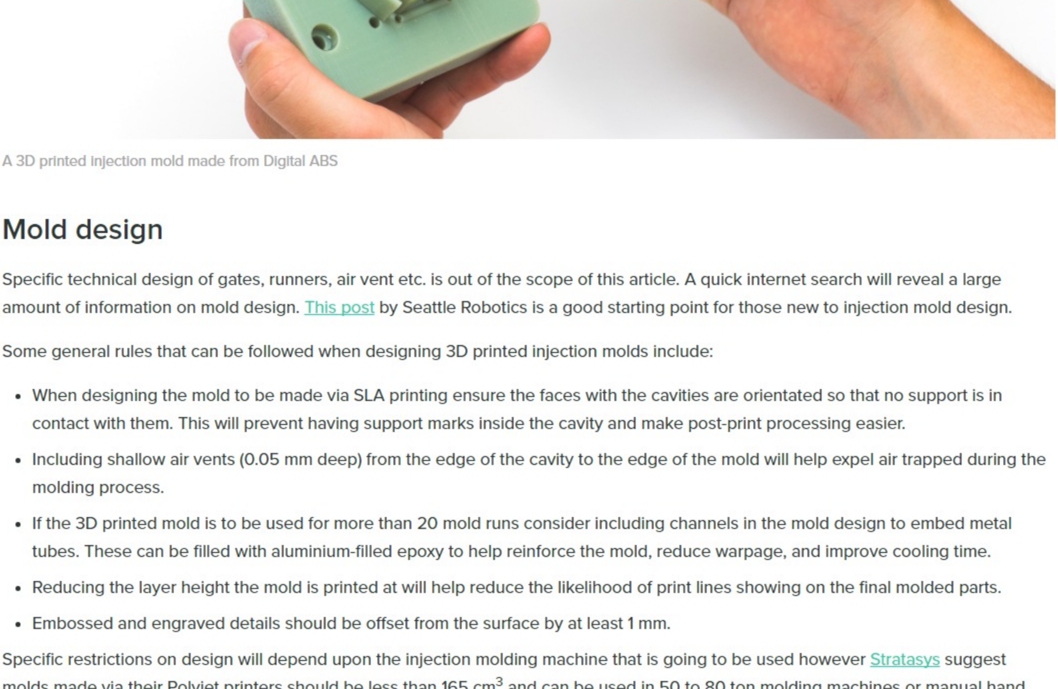
The 3 main material characteristics that will govern whether a 3D printed low run injection mold will work are:

- **High temperature resistance** - A high HDT temperature is required to withstand the molten melt that is injected into the mold and still produce accurate parts.
- **High stiffness/toughness** - Removing parts from the mold repetitively requires material that is tough with a high stiffness to maintain mold accuracy.
- **High level of detail** - One of the main requirements of an injection mold is high accuracy. High accuracy molds produce high accuracy parts.

Because of these characteristics, SLA and Polyjet are the technologies best suited for the production of injection molds. SLA and Polyjet are the 3D printing technologies that produce parts to the highest level of accuracy and ad are ideal for prints with intricate details and features. In particular materials like Formlabs High Temp resin or Stratasys Digital ABS are ideal for mold and tooling applications. An outline of the properties relevant to injection mold printing for these two materials is shown below.

Property	Formlabs High Temp resin*	Stratasys Digital ABS**
Heat deflection temperature	289 ℃ @ 0.45 MPa	92 - 95 ℃ @ 0.45 MPa
Flexural modulus	3.3 GPa	1.7 - 2.2 GPa
Impact strength (Notched IZOD)	14 J/m	65 - 80 J/m
Lowest layer height (resolution)	25 - 50 microns	16 - 30 microns
Minimum detail size	0.2 mm	0.2 mm

*Information sourced from Formlabs
**Information sourced from Stratasys



A 3D printed injection mold made from Digital ABS

Mold design

Specific technical design of gates, runners, air vent etc. is out of the scope of this article. A quick internet search will reveal a large amount of information on mold design. This post by Seattle Robotics is a good starting point for those new to injection mold design.

Some general rules that can be followed when designing 3D printed injection molds include:

- When designing the mold to be made via SLA printing ensure the faces with the cavities are orientated so that no support is in contact with them. This will prevent having support marks inside the cavity and make post-print processing easier.
- Including shallow air vents (0.05 mm deep) from the edge of the cavity to the edge of the mold will help expel air trapped during the molding process.
- If the 3D printed mold is to be used for more than 20 mold runs consider including channels in the mold design to embed metal tubes. These can be filled with aluminium-filled epoxy to help reinforce the mold, reduce warpage, and improve cooling time.
- Reducing the layer height the mold is printed at will help reduce the likelihood of print lines showing on the final molded parts.
- Embossed and engraved details should be offset from the surface by at least 1 mm.

Specific restrictions on design will depend upon the injection molding machine that is going to be used however Stratasys suggest molds made via their Polyjet printers should be less than 165 cm^3 and can be used in 50 to 80 ton molding machines or manual hand presses.

Designing parts for injection molding

As with conventional injection mold design consider:

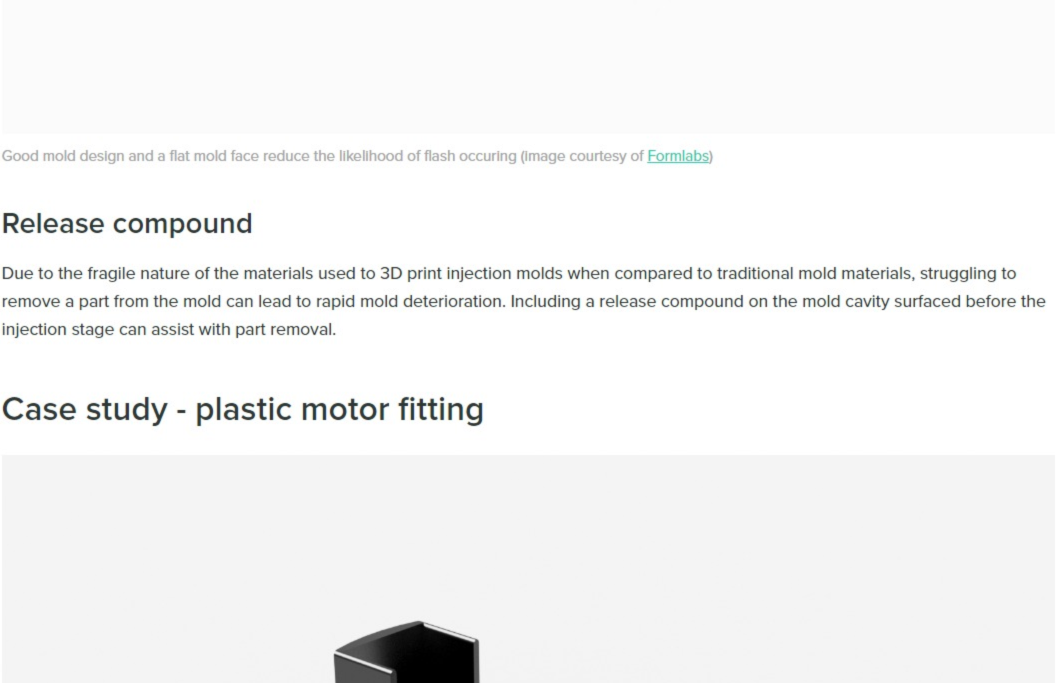
- Adding a draft angle of 1 - 3 degrees to aid in removal of the part after printing
- Maintaining a uniform wall thickness across the entire part
- Keeping all walls and features as thin as possible.
- Including radii on all edges and corners.
- Including thin ribs and gussets to add strength to a part rather than increasing wall thickness.

Draft angle design for injection molding

Reducing flash

Flash is the name given to the material that comes out between the halves of the mold during the injection process. This generally occurs when the two mold halves do not mate perfectly together, are not perfectly flush and flat or the mold is overfilled. Runners are used in mold design to help reduce the likelihood of flash occurring.

If designing for an aluminium frame, add 0.125mm of extra thickness to the back of the mold plates to account for compression forces and to ensure a complete seal. Increasing clamping force in the vise can also help mitigate flash, as can polishing the mold's split plane to give it as flat a surface as possible.

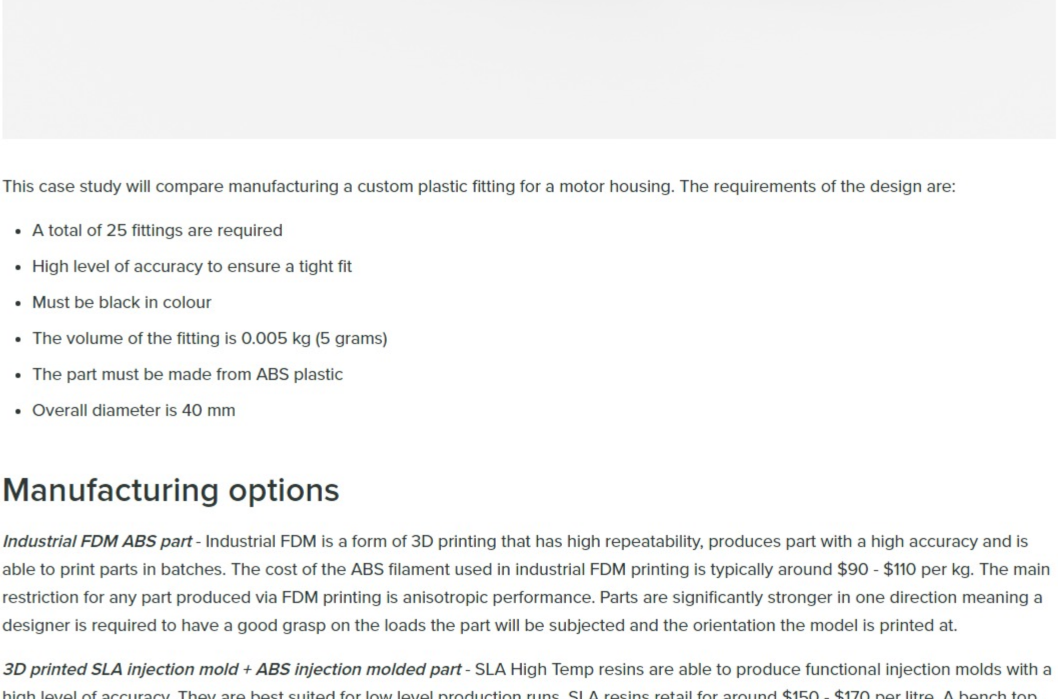


Good mold design and a flat mold face reduce the likelihood of flash occuring (image courtesy of Formlabs)

Release compound

Due to the fragile nature of the materials used to 3D print injection molds when compared to traditional mold materials, struggling to remove a part from the mold can lead to rapid mold deterioration. Including a release compound on the mold cavity surfaced before the injection stage can assist with part removal.

Case study - plastic motor fitting



This case study will compare manufacturing a custom plastic fitting for a motor housing. The requirements of the design are:

- A total of 25 fittings are required
- High level of accuracy to ensure a tight fit
- Must be black in colour
- The volume of the fitting is 0.005 kg (5 grams)
- The part must be made from ABS plastic
- Overall diameter is 40 mm

Manufacturing options

Industrial FDM ABS part - Industrial FDM is a form of 3D printing that has high repeatability, produces part with a high accuracy and is able to print parts in batches. The cost of the ABS filament used in industrial FDM printing is typically around $90 - $110 per kg. The main restriction for any part produced via FDM printing is anisotropic performance. Parts are significantly stronger in one direction meaning a designer is required to have a good grasp on the loads the part will be subjected and the orientation the model is printed at.

3D printed SLA injection mold + ABS injection molded part - SLA High Temp resins are able to produce functional injection molds with a high level of accuracy. They are best suited for low level production runs. SLA resins retail for around $150 - $170 per litre. A bench top injection molding machine has been used for this example with the 3D printed molds inserted into aluminium frames.

Traditional injection molded ABS part - Traditional injection molded parts have a very high level of accuracy, excellent surface finish and a very high level of repeatability. The main downsides to traditional injection molding is the high initial setup cost and the number of design conditions that must be implemented in the design of a part (draft angles, constant wall thickness etc). ABS pellets used in injection molding sell for approximately $2 - $3 per kg.

A summary of the prices (based on online quotes) to manufacture the ABS fitting using the technologies discussed above is summarised in the table below. All prices are excluding shipping.

	Industrial FDM*	3DP IM**	Traditional IM***
Cost of mold	-	$70.85	$1660.72
Cost per part	$3.69	$0.05	$1.89
Total cost	$92.25	$72.10	$1711.48
Lead time	4 days	2 days	8 days

* Quote sourced from 3DHubs.com with ABS, 20% infill printed on a Fortus 250MC
** Quote sourced from 3DHubs.com with Formlabs High Temp resin printed on a Form2
*** Quote sourced from Protolabs

Written by

Ben Redwood
Mechanical engineer working at 3D Hubs